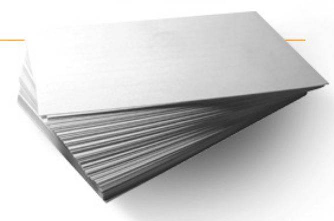

HARMONIC VIBRATION ANALYSIS

Establishing, Identifying and eliminating harmful frequencies

By Vik Vedantham, CAE Specialist, 3DVision Technologies

Any design that is subjected to a cyclical excitation source responds by vibrating in order to release the incoming energy. The vibrations are perceived as small deflections on the design. Everyday instances include motors mounted to shafts, a tool mounted on a mill or a lathe, unbalanced rotating machinery, etc. The common denominator in such cases is that a cyclic sinusoidal frequency source induces vibrations on the design, resulting in deflections. Such deflections over time are typically detrimental to a design, although in a few cases, they are desired outputs.

Establishing the nature and response of a design to such vibration sources thus becomes an important part of the development cycle. Traditional methods such as shaker table tests or road tests have been the ways of the past to establish the response of a design. Today, the same sinusoidal loads can be applied to a design inside SolidWorks in order to understand how it will respond. Such a study can assume exponential importance in eliminating premature failure.

WHAT IS HARMONIC ANALYSIS?

A harmonic load is a cyclic load such as the composite waveform at the bottom of the adjoining picture (Fig. 1). The composite wave load that the design sees can be split up into a series of simpler sinusoidal waves (the five waves in the adjoining picture) that combine to create the complex cyclic curve.

Harmonic analysis involves understanding which of these derivatives come closest to the natural vibrating characteristics of the physical design itself (natural frequencies of the design). When the natural frequency of a design coincides with one of the derivative frequencies of the composite wave, the design goes into resonance and results in excessive deflections, stresses and can possibly hasten failure.

SolidWorks Simulation (previously known as COSMOS) can effectively perform the above procedure by first extracting the natural frequencies of a design, and then subsequently studying the response of the design close to these natural frequencies in the operating range of the excitation source. The results of the simulation would be a measure of stresses, deflection and strain across the requested operating frequency range.

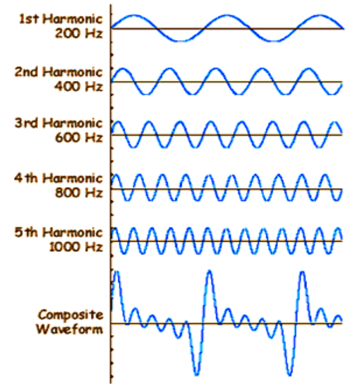

Fig. 1 A complex cyclic waveform and its constituent

CASE STUDY – ENGINE FRAME

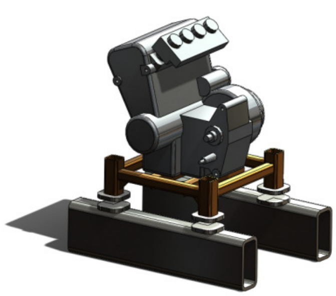

Fig. 2 Model of an engine block mounted on a frame

OBJECTIVE The model shown in Fig. 2 is that of an engine block mounted on a sub-frame and a rail that travels through multiple workstations during the manufacturing cycle. During this process, the entire unit will experience a range of constant cyclic excitations from external sources (such as operating machinery around the unit). The expected range of frequencies that this unit will experience varies from 0 Hz to 800 Hz.

The objective is to ensure that the stresses and deflections encountered by the frame in the frequency range lies within safe limits so that the design is intact. In this case, the frame should meet a Factor of Safety of 2.5, and the resulting deflections should be less than 0.02 inches. The permissible changes include either a structural change, or a material change, or both.

THE MODEL The components making up the SolidWorks Assembly include the engine block; the center frame unit with gussets and cross members; pads that attach the frame to the bottom rails; and shock isolators (springs) mounted between the pads. The object of interest here is the frame and the rails, while the engine block serves as just a dead load (which will be suppressed, and the weight of the engine will be specified as a downward force in the analysis, thus eliminating the need to mesh the engine block itself).

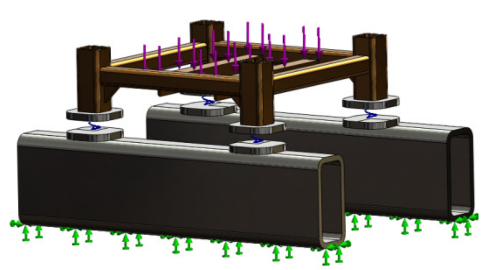

Fig. 3 Boundary conditions on the model

THE ANALYSIS SETUP SolidWorks Simulation offers the user the ability to easily setup the problem by interacting with the SolidWorks model throughout the setup. The user can ask Simulation to pull in the material definition from SolidWorks, understand locations of contact based upon how the assembly has been mated together, specify connectors such as bolts and springs virtually, as well as create the Finite Element mesh based on the parametric dimensions used to build the SolidWorks geometry.

In this case, the material used for all the components was Plain Carbon Steel from the SolidWorks material library. As seen in Fig. 3, the model has been constrained on the bottom rails (Green restraints), while the weight of the engine block is applied as a force on the top cross members (purple arrows going downward). The springs that represent the shock isolators have been suppressed from the SolidWorks geometry, and substituted with virtual Spring Connectors that carry the axial and tangential stiffness properties of the physical shock isolator. This step, once again, helped in simplifying the actual processing time while preserving the intent of the analysis. A Modal Damping coefficient of 0.03 was also specified, and the analysis was subsequently meshed and solved.

List Modes

Study name: Harmonic Response

Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	2868.5	456.53	0.0021904
2	3920.4	623.96	0.0016027
3	4024.9	640.58	0.0015611
4	4602.8	732.56	0.0013651
5	4792.1	762.68	0.0013112
6	4962.6	789.82	0.0012661
7	5048.5	803.49	0.0012446

Close Save Help

Table.1 The extracted Resonant Frequencies

THE RESULTS It is well established that resonance causes maximum deflections, and hence, the worst damage in any geometry. Since resonance occurs when the natural frequencies of the design match excitation frequencies, it is important to study stresses and deflections close to the natural frequencies.

The first output of the harmonic simulation helps understand the resonant or natural frequencies of the model. Table. 1 shows the resonant frequencies in the range of interest (0 – 800 Hz). It was observed that the first six frequencies were within the 0 – 800 Hz range. Thus, the focus of the subsequent harmonic response was to meet the objectives close to these six frequencies.

The stresses and deflections were observed individually close to each resonant frequency, and also across the entire spectrum (envelope plot, shown in Fig. 4). It was observed that the peak stress in the model across the entire operating spectrum of 0 – 800 Hz was about 42500 psi. The resulting Factor of Safety was found to be 0.75, which indicated that the design was going to fail under the excitation frequency range. Furthermore it was observed that the highest stresses were seen close to the 4th resonant frequency by creating a plot of the stresses across the frequency spectrum (Fig. 5). The resultant peak deflection of 0.048 inches was also higher than the required amount.

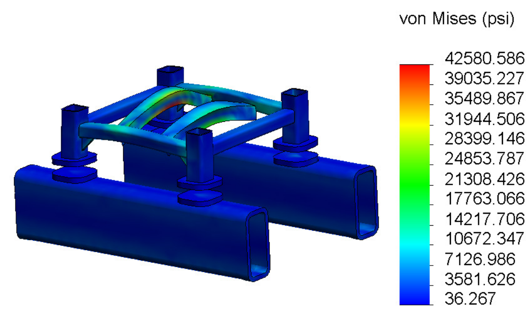

Fig. 4 Stress plot showing high stress locations

Since the design failed under the operating conditions, the possible alterations were explored. One of them involved a material change. Upon switching the material from Plain Carbon Steel to a stronger material (AISI 1020), it was seen that only two natural frequencies fell inside the 0 – 800 Hz belt. The peak stresses also came down to 8000 psi, which made the model extremely safe (FOS of 6.375). The maximum deflection was observed to be 0.005 inches.

The mass of the frame underwent a subtle change from 30.31 lbs to 30.49 lbs during the material change. However, since the factor of safety indicated that the design is extremely safe, it was obvious that material could be removed. The mass of the frame came down to 30.01 lbs by changing the profile of the four side members to a C-Channel, and by reducing the thickness of the angle irons.

Upon re-running the simulation with the lighter design and the new material choice, it was observed that the FOS came down to 3.3, while the deflections were about 0.0125 inches.

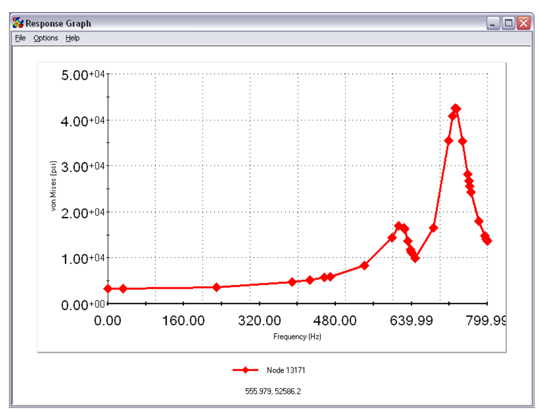

Fig. 5 High stresses close to the higher frequencies

CONCLUSIONS The ability to simulate the real-life excitations on the design helped in understanding the response of the initial design to the excitation forces. The original design would have failed under field loading conditions. The ability to perform quick structural as well as material changes inside SolidWorks Simulation helped in arriving at the final design that would meet the working requirements of the frame. Harmonic analysis, thus, formed an integral part of the developmental cycle for this fixture design.